FSI-87-TN5

Technical Note No. 5

PLUME RISE IN A STRATIFIED FLUID

C.W. Hirt and R.P. Harper
Flow Science, Inc.
October 1987

I. The Problem

The evolution of a buoyant plume in a stratified fluid is a flow problem that occurs in many natural and man made situations, for example, volcanoes, industrial smoke stacks, and hot water discharges in thermally stratified ponds. In this note we model the injection of a light fluid into a tank containing a second fluid that is stably stratified.

The tank is a cylinder of radius 35 cm and height 100 cm. The top half of the tank (50 cm) has a uniform density of 1.011 gm/cc. In the lower half (50 cm) the density increases linearly from this value to a maximum value of 1.18 gm/cc at the tank bottom. At the center of the tank floor a second fluid of density 0.848 gm/cc is injected vertically upward from a nozzle of diameter of 1.0 cm. The velocity of injection is 200 cm/s.

The distribution of injected fluid as time proceeds depends on the extent to which this fluid mixes with the ambient fluid in the tank. If, for example, the fluid is injected slowly enough that it does not mix, then it should simply rise straight up to the top of the tank because of its low density. However, if it mixes significantly with the ambient fluid near the injection point, then the average mixture density may become greater than the density of ambient fluid at some higher elevation. If this should happen, the injected fluid will reach a limiting height and then spread out horizontally. Clearly, the limiting height depends not only on the initial densities in the tank and the injected fluid, but also on the rate and degree of mixing of the two fluids. At high injection velocities one would expect significant turbulence mixing to occur. For the present case the Reynolds number of the injected flow is about 20,000. This relatively high value, coupled with large shear and buoyancy forces near the injection point, suggests that a good deal of mixing is likely to exist in this example.

The Computational Model

FLOW-3D was used to model the plume and tank as a two-dimensional, axisymmetry flow problem. The code's two-fluid option was selected so that the injected fluid could be tracked separately from the ambient fluid. To model the stratification of the ambient fluid (fluid number 2) we used the thermally buoyant flow option with a temperature distribution and thermal expansion coefficient that produced the desired initial density stratification in the tank. No heat conduction was allowed (i.e., zero conductivities), so we can think of the stratification as produced by salt or a mixture of two fluids with a vanishingly small diffusion rate.

The injected fluid (fluid number 1) was assumed to have a constant density, so it was assigned a thermal expansion coefficient of zero. The injected fluid was also given a thermal capacity one thousand times lower than the ambient fluid so that its (negligible) heat capacity would not significantly change the temperature, hence density, of the ambient fluid.

Kinematic viscosities of the two fluids were taken to be equal to that of water, 0.01 cm^2/s. Because we expect turbulence to be a significant feature of the flow, we also exercised the two-equation turbulence model included in the FLOW-3D options. This model has turbulence energy production terms arising from buoyancy effects, which are likely to be important in this problem. Boundary conditions for turbulence energy and turbulence dissipation at the nozzle injection point were chosen to correspond to a 1% turbulent kinetic energy level in the incoming flow with an integral scale of turbulence equal to one-tenth the nozzle diameter. Initial values of turbulence in the mesh were at the minimum default level, which gives a turbulent viscosity equal to the molecular value of viscosity.

All constants in the turbulence transport equations were left at their default values. In making comparisons with experimental data it may be necessary to adjust these values somewhat, especially those associated with the buoyancy terms.

In keeping with the expectation of turbulent mixing we turned off the sharp interface tracking option. This allows the two fluids to undergo turbulent mixing; the ratio of mass to momentum diffusion rates was taken to be unity.

Laboratory experiments corresponding to this physical arrangement are typically performed with a free fluid surface at the top of the tank. In FLOW-3D we cannot represent both a two-fluid interface and a free fluid surface. Since the two-fluid interface was of particular interest (to track the injected fluid), we replaced the top surface with a constant pressure boundary. This added one additional mesh cell (of 7 cm height) to the region modeled. The top boundary pressure was set to zero so that all computed fluid pressures can be interpreted as gage pressures.

An initial hydrostatic pressure distribution was assumed to exist in the tank with the fluid at rest. Gravity was a standard 980 cm/s^2 acting in the negative z-direction. To model the nozzle injection a specified velocity boundary condition was used at the bottom of the computational mesh with a thin obstacle used to close off all of the boundary except for the nozzle opening. The obstacle extends only 0.01 cm vertically into the tank so it does not block any significant portion of the tank.

Computational Results

Figure 1 shows the mesh chosen to model the tank and nozzle arrangement. A total of 390 cells is used to define the cross section of the tank. Finer zoning is used at the axis of symmetry (left side) in order to resolve the nozzle. Only one mesh cell of 0.5 cm is used to define the nozzle radius. This is poor computational practice, in general, but it is dictated by the small size of the nozzle compared to the tank dimensions. Finer zoning could be used to further resolve the nozzle injection region if the details needed there justify the extra computational cost. For this sample case we note that even with a one cell resolution we do have the correct mass and momentum injection rates.

Initial pressure contours are shown in Fig. 2. Although there appears to be a uniform hydrostatic pressure distribution, careful measurement of the spacing between the contours shows that there is a slightly larger pressure gradient in the lower half of the tank containing stratified fluid.

Figure 3 contains plots of the developing flow after 2.0 s. The velocity plot reveals little information because the flow is largely confined to the inlet region at the lower left corner. The contours of fluid fraction show the distribution of injected fluid (i.e., fluid number 1 volume fraction). Note that the maximum value is at the inlet, but is only 0.626, which indicates that considerable mixing occurs immediately. In the absence of mixing the maximum value would be 1.0. The overall hydrostatic pressure distribution has hardly been disturbed.

A better view of the flow at this early time can be obtained by plotting just a small portion of the computational region near the lower left corner. For example, Fig. 4 shows an enlargement of the first 3 radial and first 11 axial mesh cells. Here the characteristics of the plume are more clearly visible. In the velocity plot the line circling the lower three velocity vectors is a contour line indicating equal portions of fluids 1 and 2.

After 10 s the flow has developed as shown in Fig. 5, which again shows the situation in the entire flow region. Except for the contours of fluid fraction and density, there is little difference to be seen in comparison with the results at the earlier time. In Fig. 6 are the corresponding 10 s plots for the 3 by 11 cell region near the nozzle. These plots are almost identical to those at 2 s, which indicates that the flow near the inlet is nearly in a steady-state condition.

As time proceeds the principal effect is to push more plume material to higher altitudes where it exhibits an interesting behavior. The plume center rises to a height of about 42 cm, but most of the plume material is spreading radially outward at an average height of 25 cm. To see the flow in this region more clearly refer to Figs. 7-9, which are plots containing only 12 axial cells starting from the 7th cell above the tank floor. In Fig. 8 the length of the velocity vectors has also been increased by a factor of 8 (over the plot default size) to emphasize the flow responsible for the radial expansion of the plume.

Comparing these plots we see that the inertia of the plume is carrying relatively dense fluid up to altitudes where the ambient fluid is less dense, see Fig. 7. Negative buoyancy effects eventually overcome the inertia and the fluid then falls down and outward, Fig. 9. Close inspection of the velocities in Fig. 8 suggests that there is an oscillatory up and down motion to the flow as it moves out radially. This is reasonable since the fluid falling down from the center of the plume overshoots its equilibium height in the surrounding stratified fluid. In fact, associated with the plume's lateral expansion there should be internal gravity waves in the ambient fluid, but the limited resolution and size of the tank prevent this feature of the flow from being clearly shown.

The calculation was terminated at the 10 s point since it is evident that at later times the principal change in these results will simply be further radial expansion of the plume material.

Total calculational time to reach the 10 s level on a MicroVAX II computer was 13.89 hours. Time-step size was limited to 0.00166 s by turbulent diffusion processes in a region approximately 5 cm directly above the nozzle. This is the region where the dynamic turbulent viscosity is greatest, as shown in Fig. 11.

In retrospect, we could have eliminated much of the upper portion of the tank from the analysis since the plume only rose about half way. At early times we could also reduce the radial region modeled and use this savings in computational effort to increase resolution near the injection region.

Summary

By selecting the proper options and physical parameters available in FLOW-3D we have successfully modeled the evolution of a buoyant plume in a stratified environment. The plume behavior is consistent with physical intuition, showing a variety of realistic features. Comparison with actual experimental data must now be undertaken to see how well the turbulence parameters and initial conditions are correctly modeling true physical mixing levels.

The input data necessary to set up this calculation for FLOW-3D is quite small, as shown in Fig. 10. The only feature that could not be defined by input data was the initial linear temperature profile used to generate the stratification. For this we programmed in a short patch in the SETUP routine that defined the initial temperatures we wanted.

Because FLOW-3D is a fully three-dimensional flow analysis program, the same physical model used in this example could also be used to investigate plumes in cross flows, the interactions of multiple plumes, or the behavior of plumes in the presence of obstacles and uneven terrain.

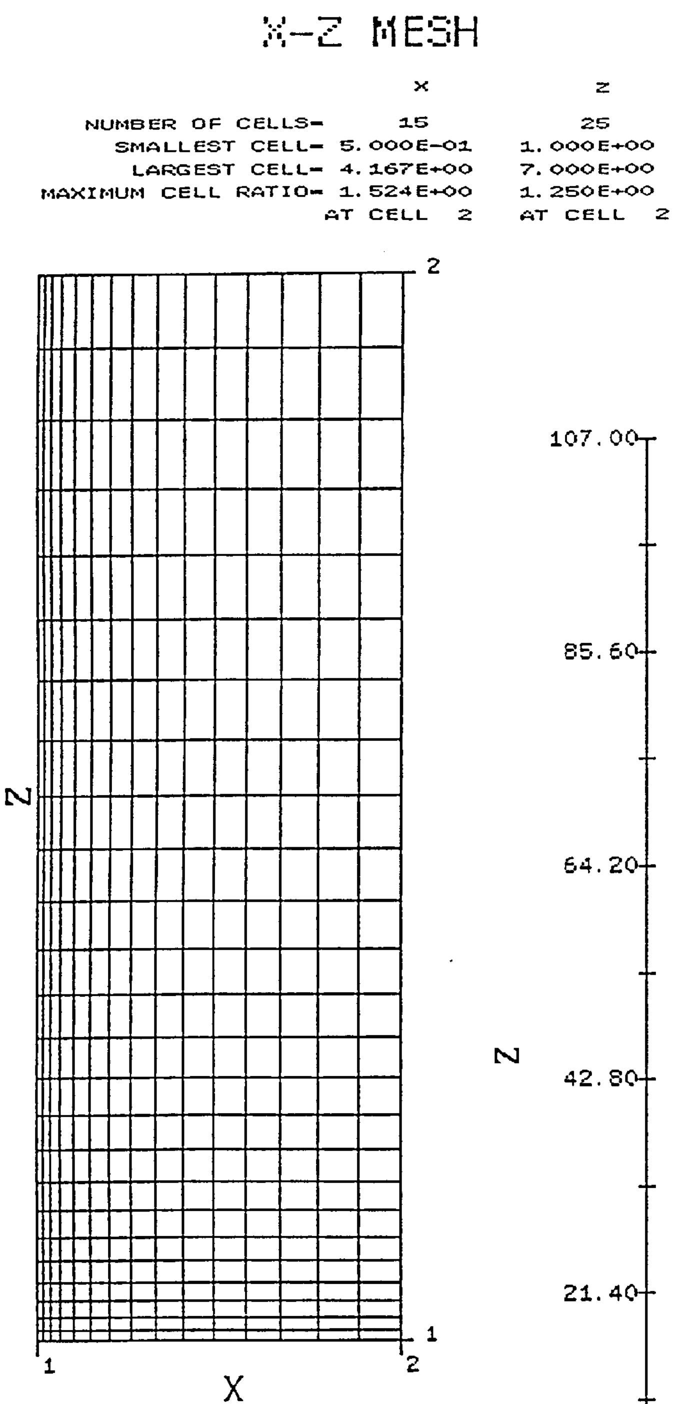


Fig. 1.

Mesh arrangement. Cylindrical axis coincides with left edge.

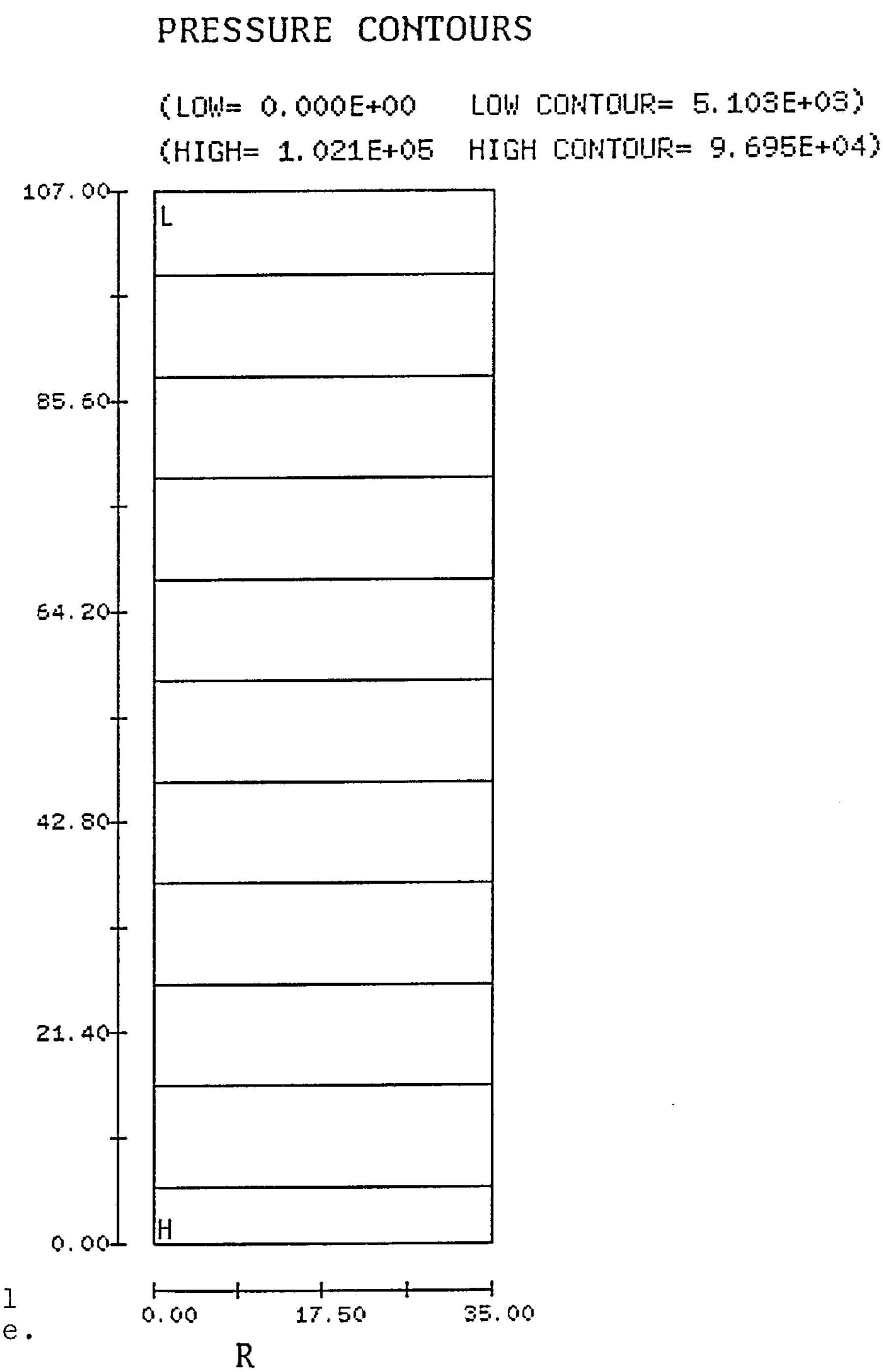


Fig. 2.

Initial hydrostatic pressure contours.

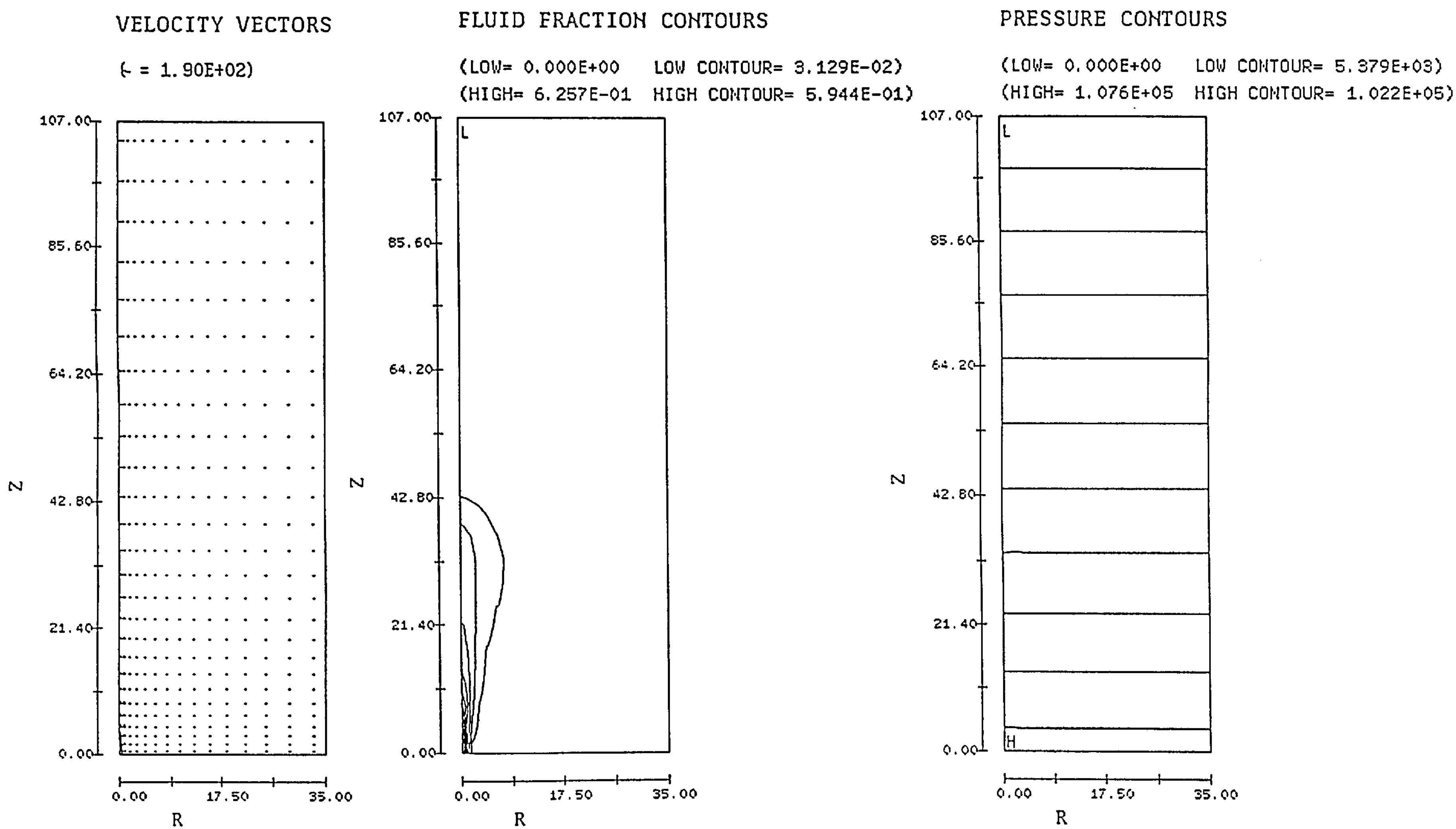


Fig. 3. Computed results at t=2.0 s.

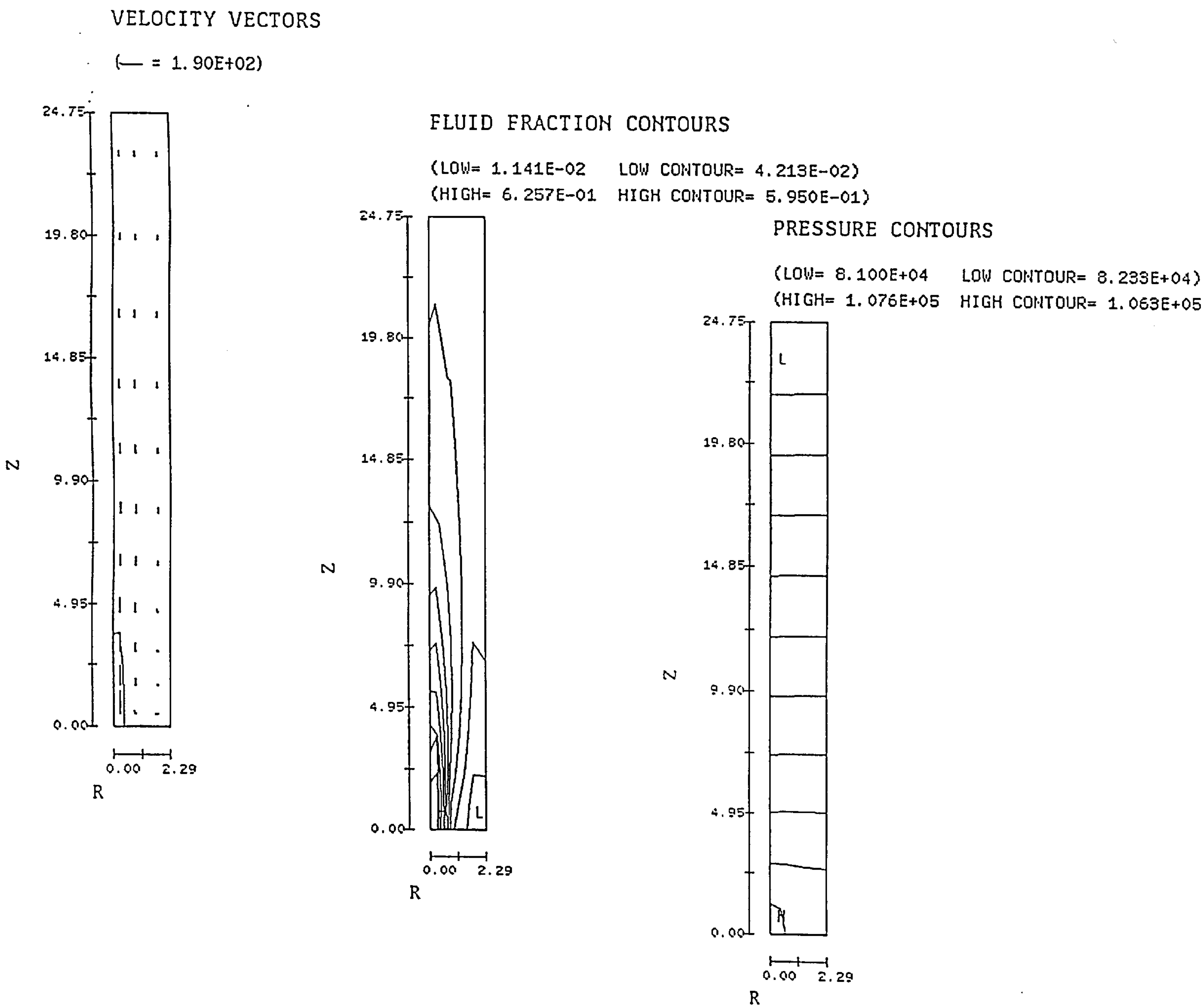


Fig. 4. Enlargement of t=2.0 s results near injection region.

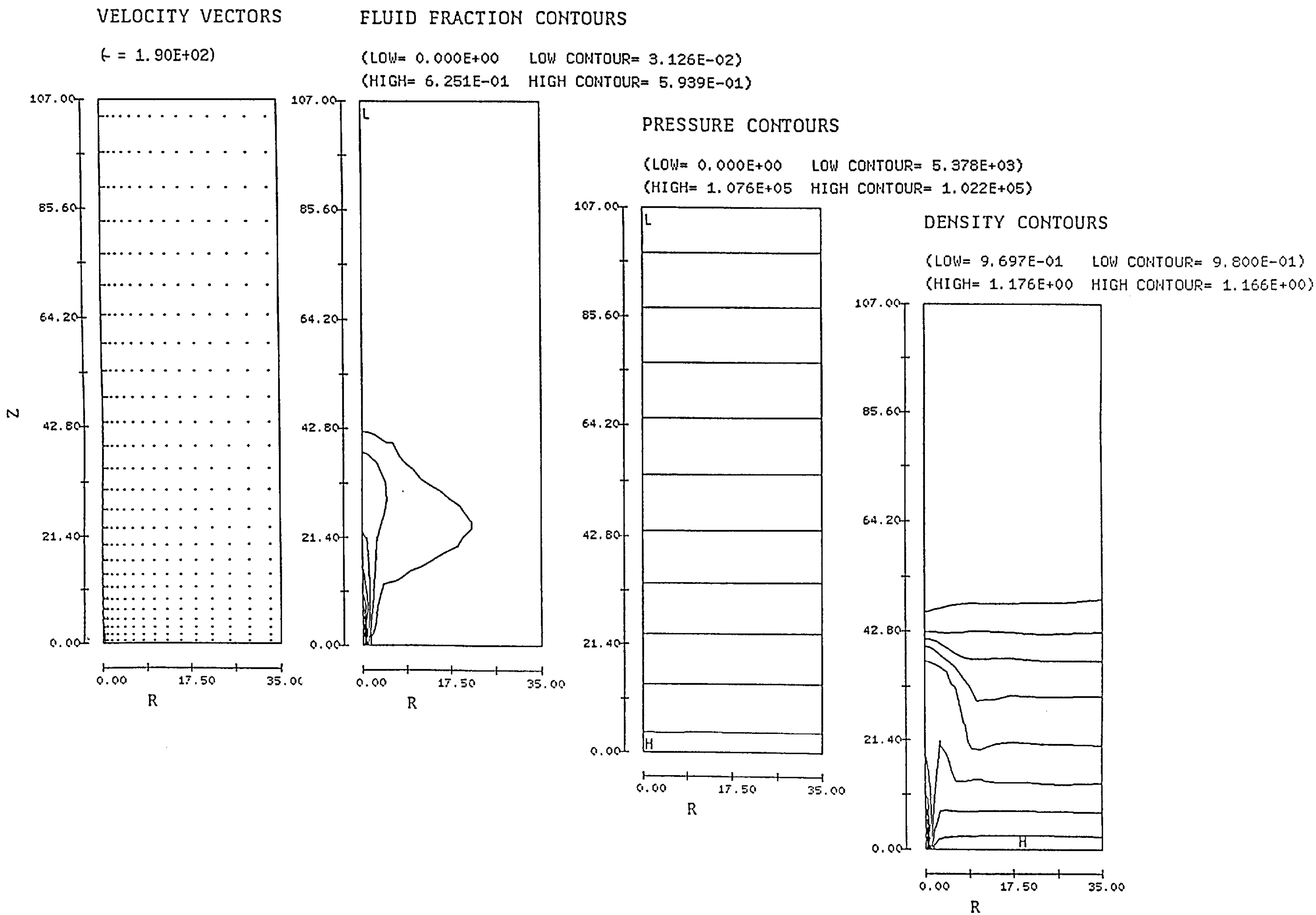


Fig. 5. Computed results at t=10.0 s.

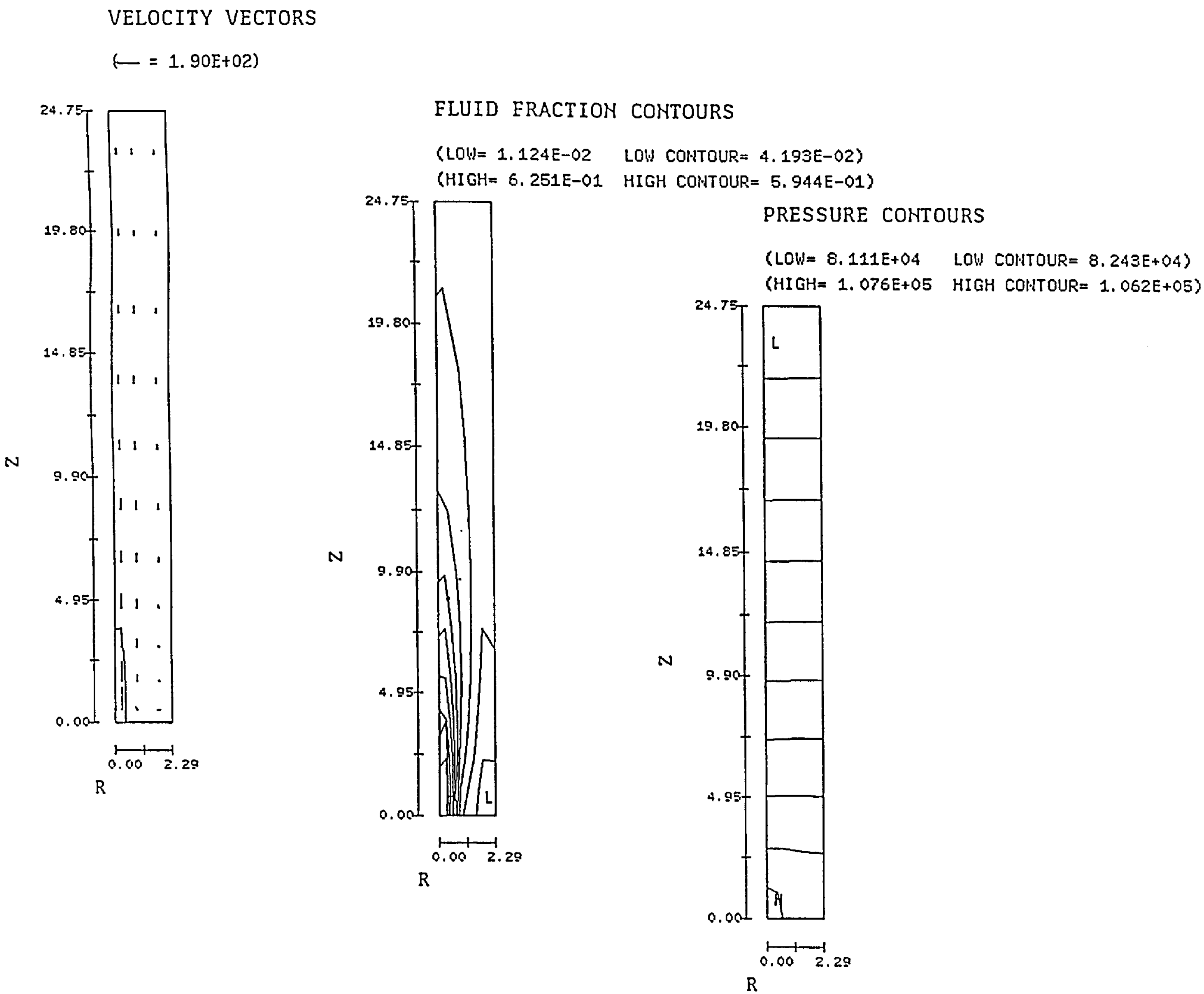


Fig. 6. Enlargement of t=10.0 s results near injection region.

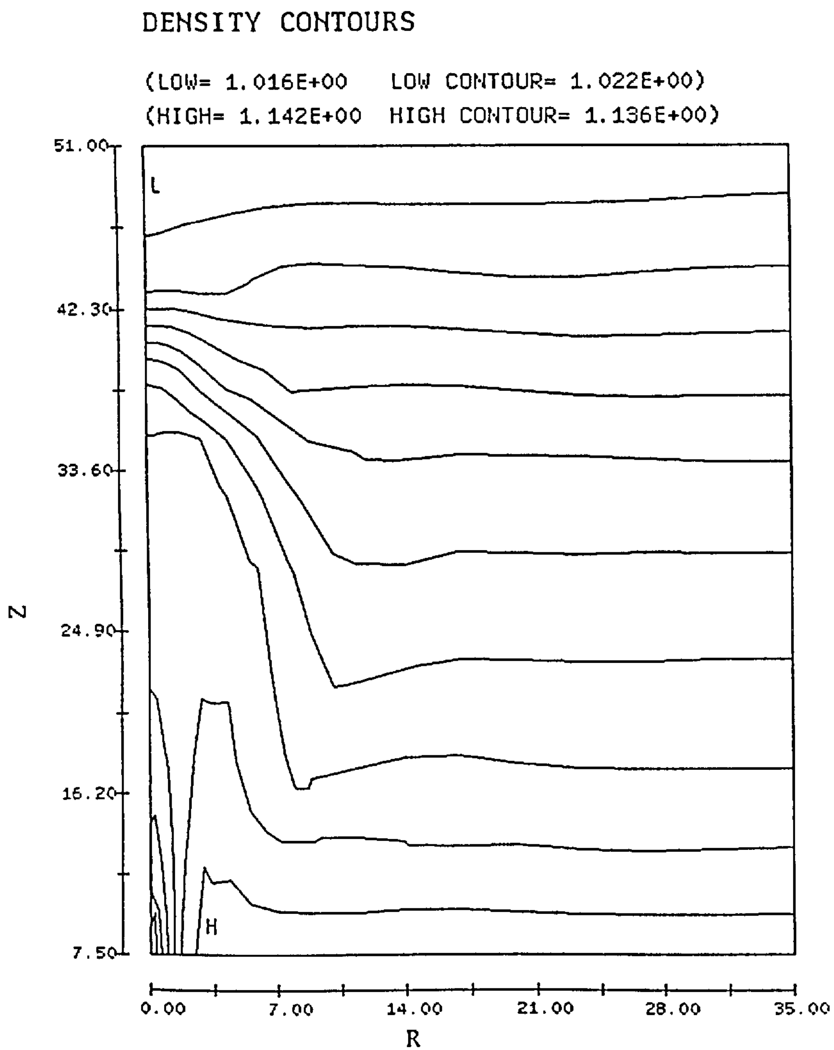


Fig. 7.

Enlargement of density contours at t=10.0 s in the central plume region.

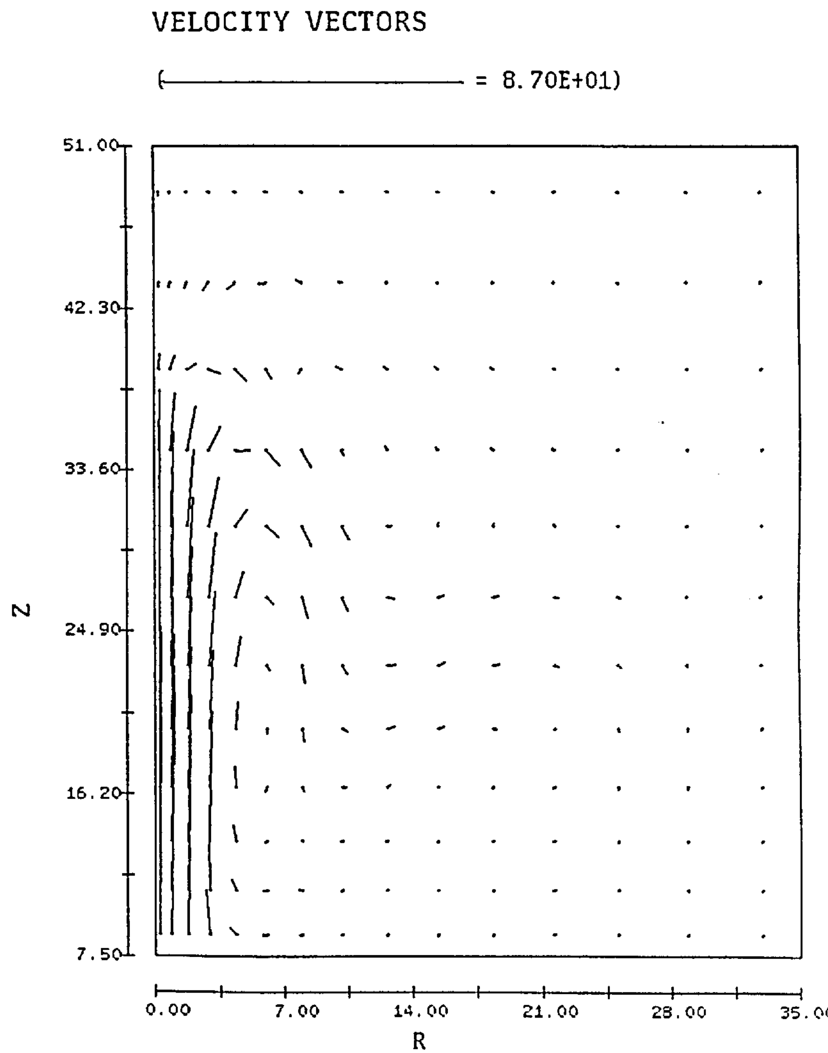
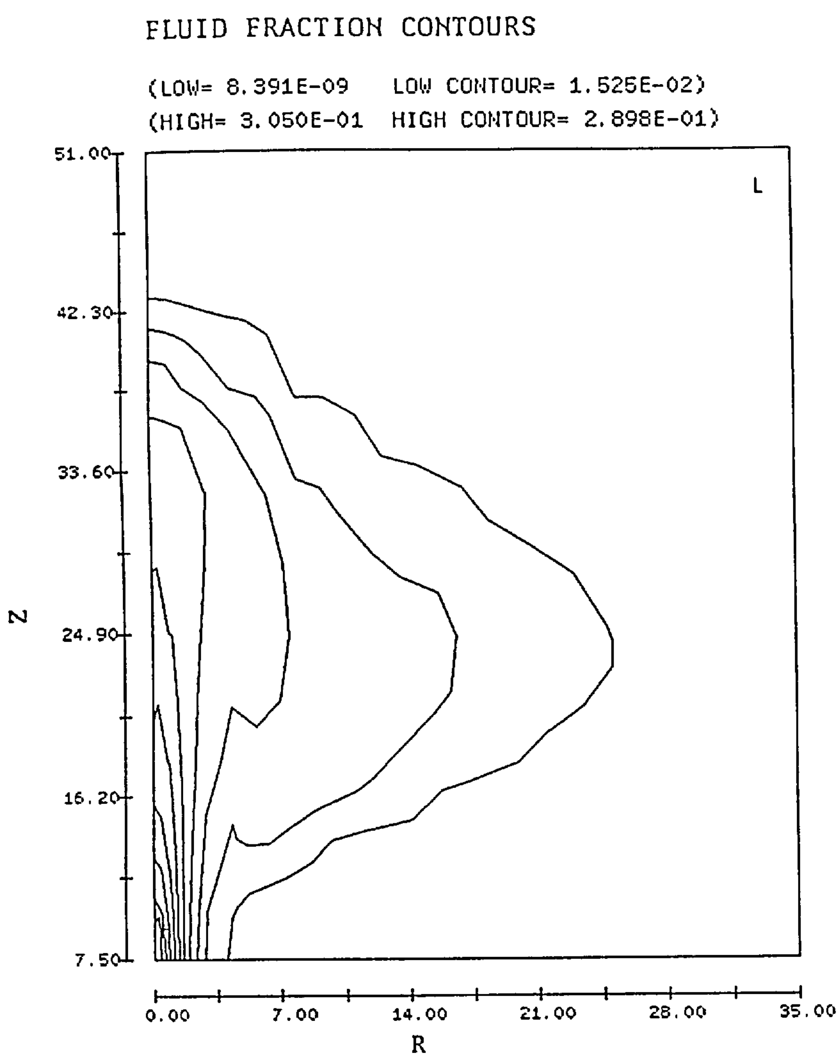


Fig. 8.

Enlargement of velocity vectors at t=10.0 s in the central plume region.




```
      PLUME
$xput
    ipdis=1,              irpr=5,               ktpr=5,
    nmat=2,               itb=0,                cyl=1.0,
    con=0.45,             ifrho=1,              delt=0.002,
    ifvis=3,              twfin=2.0,            ifenrg=2,
    iadix=1,              gz=-980.,             mui=.01,
    cvl=.001,             cv2=1.0,              rhof=0.848,
    rhofc=1.011,          tstar=150.,           thexf1=0.0,
    thexf2=0.00334,       rmf=1.0,              rmdtke=1.0,
    rmrhoe=1.0,           rmtke=1.0,
    wt=5,                 pbct(1,6)=0.0,        fbct(1,6)=0.0,
    tbct(1,6)=150.,       tkebct(1,6)=1.0,      dtkbct(1,6)=9.0,
    wb=6,                 fbct(1,5)=1.0,        tbct(1,5)=150.,
    tkebct(1,5)=200.,     wbct(1,5)=200.,       dtkbct(1,5)=3.12e+3,
$end
$mesh
    px(2)=35.,            sizex(1)=0.5,         nxcelt=15,
    py(2)=35.                                   nycelt=1,
    pz(2)=100.,           sizez(1)=1.0,         nzcelt=25,
$end
$obs
    nobs=1,               cc(1)=-0.01,          cz(1)=1.0,
                          ra0(1)=0.5,           rah0(1)=1.0,
$end
$fl
    flht=0.0,
$end
$bf
$end
$temp
    tempi=150.,
$end
$grafic
    ncplts=2,             kontyp(2)=2,          nvplts=1,
$end
$parts
$end
```

Fig. 9

Enlargement of injected fluid fraction at t=10.0 s in the central plume region.

Fig. 10.

Input file for plume problem.

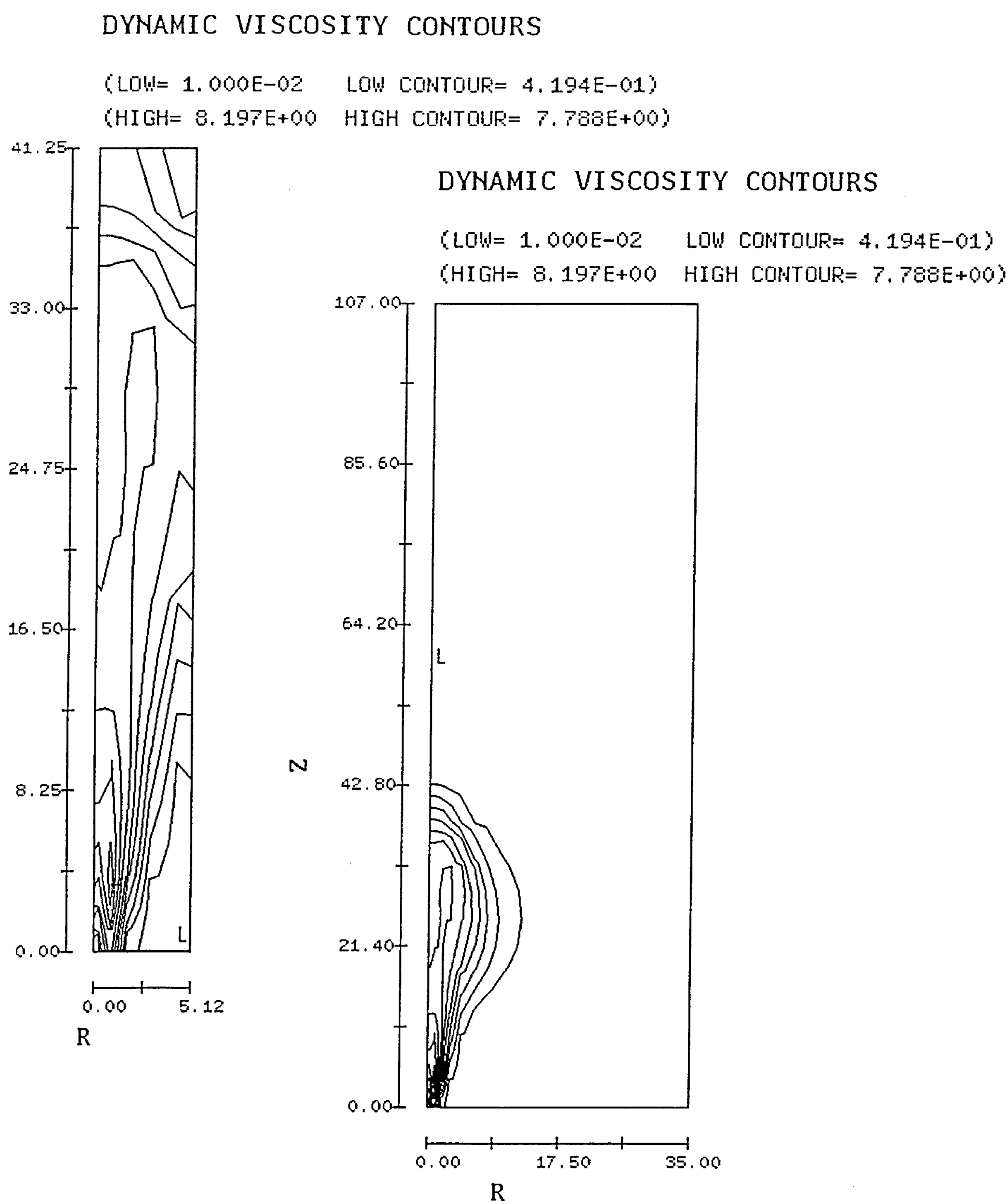


Fig. 11. Turbulent dynamic viscosity contours at t=10.0 s. Full mesh is shown on right and blowup of injection region on left.